

Machining Polyurethanes: Introduction

Cast polyurethanes can be readily turned, sawed, drilled, ground, or milled. These and other secondary operations present many similarities to the machining of metal, but there are also some important differences. This paper is intended to provide some general guidelines for machining urethanes, and also focus on the most common group of machining operations and discuss some specific tools and techniques.

It is important to note the material presented here is a starting point. The wide variety of urethane compounds and their respective physical properties and characteristics creates a wide range of machining situations. Experimentation and experience will tell you what speeds, what feed rates, and what types of tools will work best for the urethanes you machine regularly.

Harder urethanes – 90A and up – have a high degree of machinability. Lathe turning, fly-cutting, grinding, contouring, and more are easily accomplished on conventional metal-working equipment by machinists who are familiar with procedures for handling plastics.

Some different tools and techniques are required for compounds of 80A durometer and lower. These lower modulus compounds are typically machined by knifing, grinding, and sanding. In some cases, however, they can be worked like higher modulus materials by “freezing” them in dry ice or liquid nitrogen environments.

Some Key Points to Remember in Machining Urethane

- Urethanes have much lower thermal conductivity than metals, so heat generated by cutting tools stays close to the tool and raises the urethane temperature rapidly. This heat must be controlled. Melting can occur above 400°F. In addition to possible melting, heat generated by machining causes the part to expand. When that part cools, it shrinks down and can end up undersize.
- Elastic memory - Elastic recovery occurs in urethane both during and after machining. The cutting tool must provide clearance to compensate for this. With compensation, expansion of the urethane as it passes the tool will result in increased friction between the cut surface and the cutting tool. Excess heat build-up will result. Elastic recovery after machining can result in smaller internal diameters and larger external diameters than were measured during cutting.
- Modulus of elasticity - Urethanes are resilient and can easily be distorted. It is possible to alter the shape of a urethane part by clamping or chucking it with too much force. This would cause the final machined shape to be distorted after the cut had been made and the fixturing pressure was released. Care must be taken to hold parts securely, but avoid distortion due to holding or cutting.
- Softening point - Gummy, poor finishes, and poor dimensional control will occur if excess heat is generated and allowed to accumulate. Proper tool geometry, feed rates, and cutting speed, in conjunction with coolants usually overcome these problems. Water soluble cutting oils and/or light machining oils are good coolants for urethanes.

Machining Polyurethanes: Sawing & Shearing

One of the best machines for sawing urethanes is a band saw. Long blades of 125 to 175 inches are desirable because they stay cooler and keep the urethane from melting. A band type that we have found to work well is a 4 tooth per inch with raker set. A raker set blade is one that has its teeth alternating to the left and right of center.

Band speeds in the range of 3200 feet per minute work well on almost all durometers. Feed rate is controlled by hand, so it is operator dependent. Any moderate hand feed will suffice, but do not force the work.

On softer urethanes, a faster blade speed helps prevent the urethane from pulling down into the cut, rubbing on the blade, and building up heat. When cutting thin, low durometer sheet stock, the work must have some support. A sheet of cardboard, for example, will help prevent the workpiece from being pulled through the table slot by the blade.

If a finer finish is needed, change to a 10 tooth per inch blade with raker set. When cutting 90A durometer and below, use a spray mist of water soluble oil (50 – 50 mix) to help keep the heat down and to improve the finish. This spray coolant is also helpful on thick cuts where feeds are slow, and on long cuts where the blade will contact urethane for a long period.

A good alternative to saw cutting thinner and lower durometer sheets is a shear. Shearing, punching, and die cutting are possible on sheet stock up to 1" thick if the hardness is low and the tools are sharp. Remember, though, as the thickness increases, so does the tendency for the cut edge to have a "dish" due to the elasticity of the urethane. Shearing and punching are also possible on 50D to 75D urethanes, but it's only practical up to 1/4 inch thickness.

Machining Polyurethanes: Turning O.D.'s & Facing

Turning, boring, and facing operations can be performed on either a turret or engine lathe. Tool configuration, geometry, and placement, as well as rpm are dependent on what hardness the urethane is and what the operation is. Another variable is feed rate – the speed of the tool with respect to the rpm of the lathe. Feed is often controlled by hand and is subject to operator judgement and “feel”.

In general, use sharp tools, high turning speed, and slow to moderate feeds (depending on hardness). Cutting tools for urethane must have sharp, carefully honed cutting edges. Sharpen tools on a honing stone for a razor sharp edge on the sides, tip, and top of your tool. We have found success with both high speed steel and carbide tools.

Tool clearances must be greater than those used for metal. The goal is to have little or no resistance as the tool travels through the urethane. The chip (material that is being cut away) should come off as a continuous strip or ribbon. A smooth surface on the top of your tool will aid in chip removal. This is very important to prevent the chip from wrapping back around the workpiece. Good chip removal is also critical for heat removal and tool life. See figure 1.

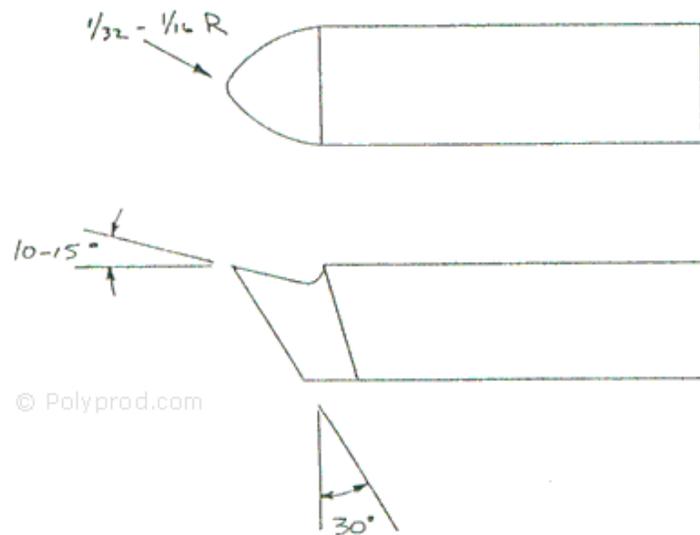


Figure 1. TURNING/FACING TOOL

Tool geometry is very important for successful machining of urethane. Rake angles and nose radius will vary depending

on durometer and desired surface finish. TOP RAKE determines chip flow. Too little causes that build-up and poor chip removal. Too much will cause reduced tool life. 10 to 15 degrees is a good starting point. SIDE RAKE is the amount of angle from the cutting edge to the bottom of the cutting tool 30 degrees is a good starting point for side rake. This angle affects surface finish. Too little allows the tool to rub against the workpiece. Too much will shorten tool life. NOSE RADIUS is the radius of the top edge of the tool at the tip. Nose radius is the most important part on a urethane tool because it significantly affects surface finish. As a rule, as durometer goes up, nose radius increases.

High durometer urethanes (95A and up) can be turned very easily. Smooth surfaces can be achieved on heavy as well as light cuts, so roughing cuts are seldom required. Surface finish is best when your removal is .050" or more so your cutting tool gets a good “bite”. We have found that on the harder urethanes, a speed of 600 to 1000 rpm works well. Feed rate depends on the desired surface finish. Typical feeds are in the .005 to .010 in./sec. range. The faster the feed rate, the more of a “record effect” you will have in your surface finish.

For higher durometer urethanes, a round-nose tool is a good choice. The radius on the cutting end affects surface finish. The nose radius should be about 1/16 with a 10 to 15 degree top rake and a 30 degree rake around both sides.

Medium hardness urethanes - 80A to 90A - require a tool with a smaller nose radius of 1/32 or less. Side and top rake should stay about the same. RPM should be faster than what is used for harder durometers - in the range of 1000 to 1500 rpm. Feeds are generally faster - in the range of .050 to .150 in./sec. This is a rapid “plunge” type feed.

When turning large diameter parts, cuts of 1/10 to 1/8 inch deep and a light feed of .003 to .007 inches per revolution are recommended. Remember also that centrifugal forces on large, low hardness parts can cause a “flaring” type deformation at high turning speeds. This effect can cause the part to be machined improperly if it is not compensated for.

Machining of Polyurethanes: Parting

Another lathe operation that is a little different than turning o.d.'s and facing is parting. Parting, or cut-off, is used to remove over-pour on higher hardness parts that are too hard to knife cut.

Tools that work well for parting are .060" to .100" wide with a 20 to 30 degree front rake and no top rake. A small 3/64 radius is ground into the top of the tool. Starting on the cutting edge, some side clearance is helpful. 3 to 5 degrees is all that is usually needed. See figure 2.

The proper tool, feed, and speed allow the chip to exit the cut with little resistance and heat build up. Parting yields a good surface finish and is a useful variation for facing certain urethane parts.

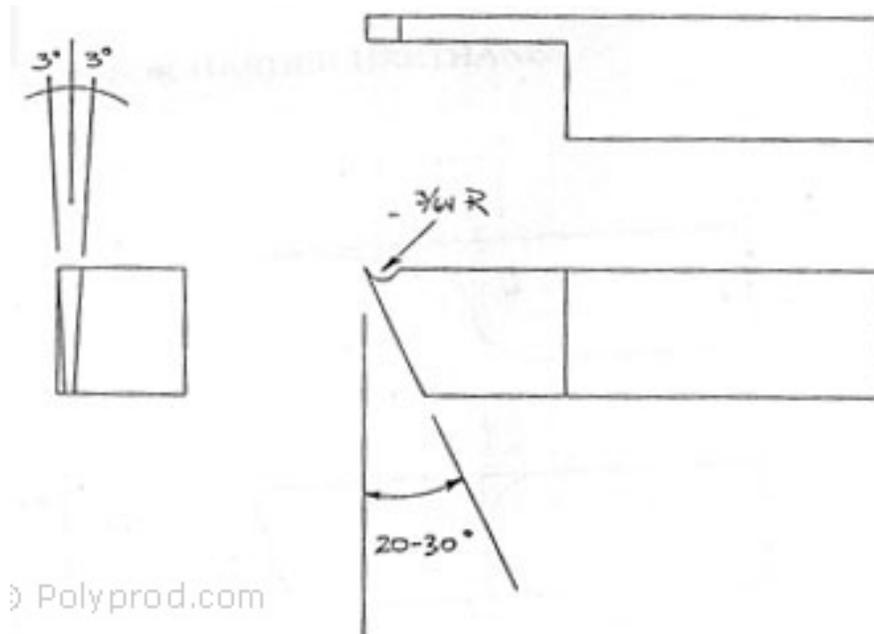


Figure2 TYPIC

Machining of Polyurethanes: Knifing

Knife cutting urethane to close tolerances can be done without too much difficulty. The tool must be absolutely razor sharp and be as thin as possible. When knifing, the urethane will have the tendency to pull into the tool. This displacement of material will cause a "dish" on the finished end of the part. The thinner the tool, the less the pulling effect on the cut edge.

A good practice on thick cuts is to do a rough cut to remove the bulk of the material, then take finishing cut to remove the final .025 to .050 of an inch.

The type of fixture used to hold the part is very important. Parts with metal inserts are usually easy to hold. Solid urethane parts can deform if they are held too tightly. Again, this would make it impossible to get a flat cut.

We use 2 types of knifing tools. High speed steel is used on medium to hard urethanes – 70A to 95A. This type of tool must be very smooth and have a razor sharp point. All edges and surfaces behind the cutting point must be smooth to prevent the cut-off material from being pulled between the workpiece and the tool. High turning speeds of 600 to 1000 rpm's with rapid hand feed will yield an excellent surface finish. See Figure 3 and 4.

Carbide blanks .250" x .125" ground on a diamond wheel to a razor sharp edge and point provide excellent cuts on low durometer, hard-to-machine urethanes. Cuts as thin as .005" are possible. The tool must be as long as the cut is deep. Turning speeds of 600 to 700 rpm work well with moderate to rapid hand feeds.

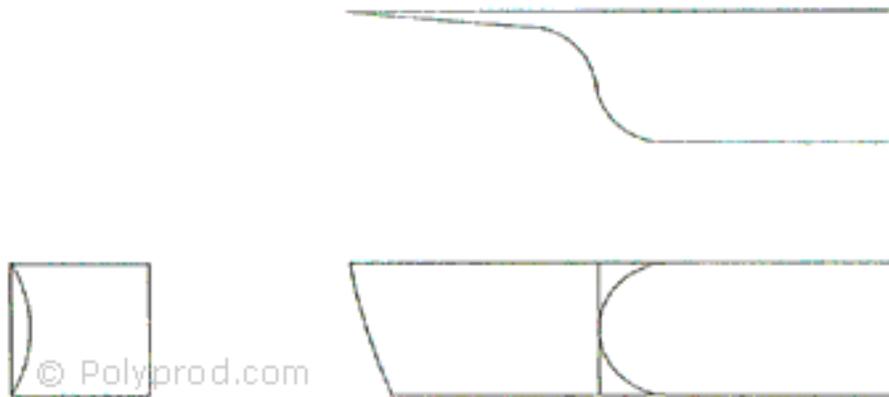


Figure 3. KNIFING TOOL FOR HARDER URETHANES

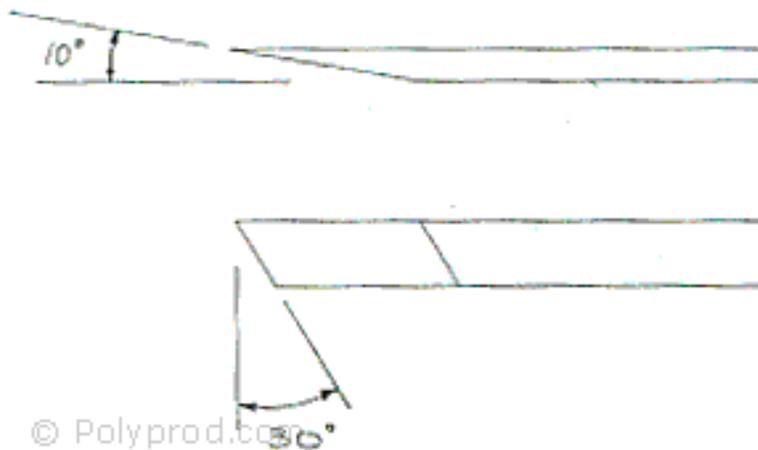


Figure 4. KNIFING TOOL FOR SOFTER URETHANES

Machining of Polyurethanes: Contouring O.D.

Machining tapers, chamfers, grooves, and other surface configurations into wheels, roller, and other round parts all fall into the general category of contouring.

Tools for grooving the o.d's of hard urethanes - 95A to 75D - all have the same basic tool geometry. We have found that a 10 to 20 degree front rake works well with no top rake. Use high rpm (depending on the diameter of the work piece) and moderately fast hand feeds.

The tool works best when positioned .025" to .075" below the center of the work piece. The chip should come off in a continuous ribbon. Try to keep the chip from wrapping around the work piece.

Large tools work well on hard durometers, but your machine must be heavy and have the horsepower to maintain constant rpm's during the cutting cycle. See figure 5.

Fixturing of parts when contouring is very important. It is possible to pull workpieces out of the machine if your fixturing isn't rock solid. When possible, pinch the workpiece between the chuck and the tailstock. Another possibility would be to hold the workpiece on a fixture in a lathe chuck or collet by bolting a plate to the fixture and sandwiching the part between the plate and the fixture. This type of fixturing is especially good when cuts are heavy, "plunge" type.

Contouring and chamfering urethane wheels and rollers is done with tools that are ground and radiused to the specific dimensions of the desired final shape. For 95A to 75D durometers, the contouring tool needs no top rake. Coolant applied with a brush will help the surface finish. The work must be kept wet during the entire cutting cycle. Remember: most contouring tools have large cutting surfaces that build up heat. This heat must be kept under control.

When contouring urethanes softer than 95A, use a tool with 15 to 30 degrees of top rake. Lathe speed depends on the o.d. of the part. In general, high RPM works best.

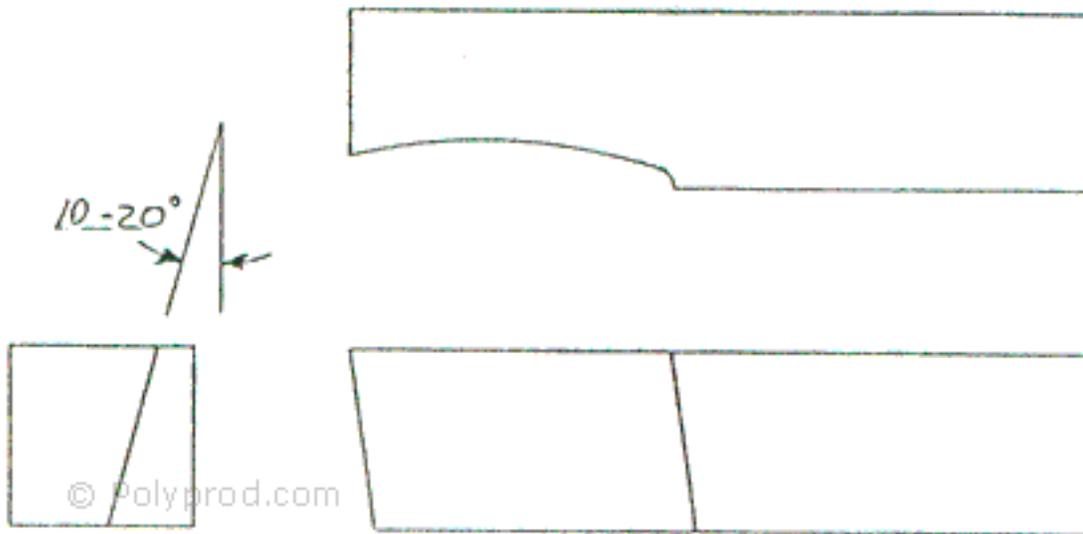


Figure 5. CONTOURING TOOL

Machining of Polyurethanes: Milling

Urethane ranging from 90A to 75D durometer can be successfully milled without much difficulty. Attempting to mill parts below 80A is not recommended. Tools must be sharp and the work must be fixtured securely.

Two-fluted end mills and single point fly cutters are preferred. Fly cutters would utilize high speed tool bits ground to a round nose. Speed of the cutter should be 900 to 1300 rpm. A feed rate of 15 to 20 inches per minute is a good starting point.

Milling is used on parts that are impractical or impossible to machine in a lathe. Milling is also used where close tolerances and a good surface finish are required.

Again, work should be held so that it is not deformed by excessive chucking pressure. Of course, work must be held securely enough so that it doesn't come loose during machining.

Machining of Polyurethanes: Grinding

Urethanes 55A to 80A durometer can be ground successfully in an engine lathe using a tool post grinder. Use low turning speeds - below 150 rpm, with the lathe running in reverse. Start with the grinder feed rate set for .005 inches per revolution. Use a slower feed rate to improve surface finish or to remove more material by taking a deeper grind.

In cases where larger amounts of material must be removed, turn the piece down with a cutting tool to within .020" of the finished dimension before grinding.

We have found that a Radiac Por-OS-Way 46 grit wheel with slight radius on the leading edge works well. RPM of the wheel should be in the 2250 to 3250 range. Again, low work-piece turning speed of 150 rpm is a good starting point. Fine abrasives can be used for final polishing.

Urethane above 80A durometer usually requires some type of coolant, however it can sometimes be ground dry. Water is good coolant, and can be applied with a brush or with a fine spray mist. Apply spray coolant on long traverse grinds. On a plunge grind where the part width is less than the wheel width, apply water with a brush to keep the urethane wet.

We recommend that the grinder be equipped with a dust collector or the operator wear an approved dust mask when grinding or sanding urethane.

Machining of Polyurethanes: Drilling

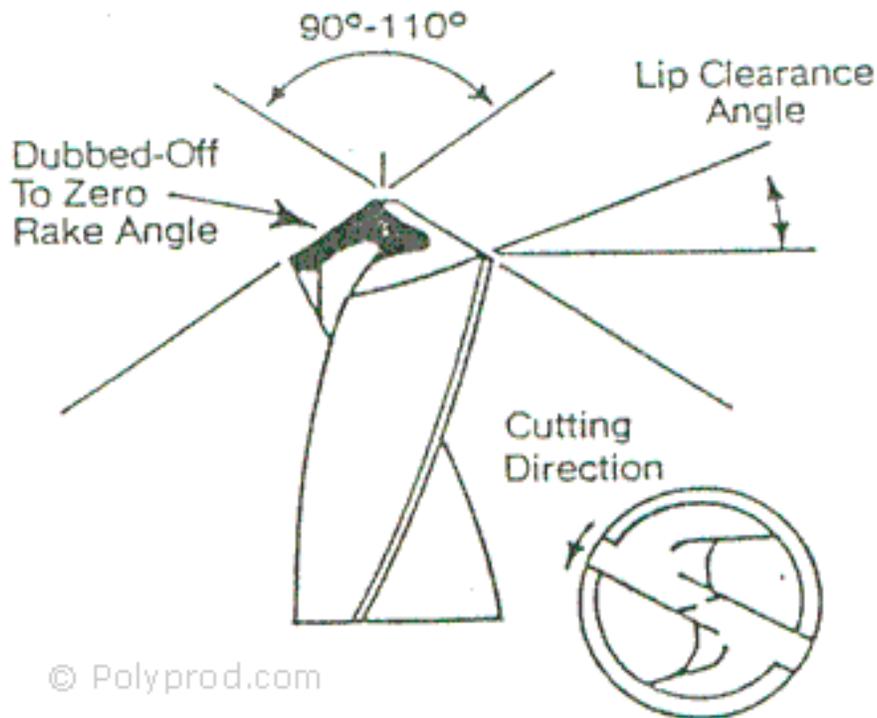
Slow, spiral drills perform best because the large flute area permits free discharge of chips with a minimum of binding and heat build-up.

Frequent retraction of the drill aids in eliminating chip blockage of the flutes. Break-out tearing at the exit side can be reduced by slowing the drill at the bottom of the hole, or by backing with another material.

When drilling a series of small holes, inserting a pin in each completed hole prevents the force of the drill from pushing material into adjacent holes and causing subsequent distortion.

Sharp cutting edges will minimize elastic deformation as the chip is formed. Polished flutes should be used to aid in chip clearance and coolant is required for good drilling performance. The rake angle should be reduced to 0 degree or negative angle and a generous lip clearance, (approximately 16 degrees) provided for proper relief.

The point angle is governed by the final wall thickness. Sharp points of 90 to 110 degrees are best for heavy walls and large diameters, while blunt angles of 115 to 130 degrees are better for thin walls. Close tolerances call for feed ranges of 0.004 to 0.006 ipr. Feed rate of 0.015 ipr can be used where tolerances permit.



© Polyprod.com

Figure 6. TOOL FOR DRILLING

Machining of Polyurethanes: Safety Considerations

When performing any machining or other secondary operations on urethane, we recommend that all appropriate safety equipment as well as personal protective devices be utilized at all times. To fully address all of the safety issues applicable to a machine shop is beyond the scope of this paper. Good judgement and a thorough understanding of machining procedures is essential.

Eye protection should consist of approved safety glasses with side shields or full goggles. A full face shield is recommended whenever chip pieces or workpieces could fly out and strike the operator in the face.

In our coverage of fixturing and chucking of workpieces, we said too much hold pressure can distort the urethane and cause machining errors. Fixturing is an extremely important aspect of machining urethane. Never attempt to start

machining until you are positive that the fixturing is safe and secure. Improper fixturing during plunge cuts and contouring can cause the workpiece to come out of the machine and injure the operator or a bystander.

Loose machine parts, handles, hand tools, etc. should not be left on or near a machine during operations. If a ribbon-type urethane chip wraps back around the workpiece, these loose items can be caught or thrown and cause a serious injury.

CAUTION ! Excessive heat can be generated by improper machining practices. If smoke is generated by machining, the method must be immediately corrected. **DO NOT** inhale the smoke or grinding dust from urethane or any elastomer.

When grinding or sanding urethane and generating dust, a dust collector should be utilized. If this is not possible, the operator should properly don and wear an approved dust mask and be sure that a good face seal is achieved.